Energy Conversion and Management 142 (2017) 414–425

Contents lists available at ScienceDirect

Energy Conversion and Management

journal homepage: www.elsevier.com/locate/enconman

Turbulence model-free approach for predictions of air flow dynamics and heat transfer in a fin-and-tube exchanger

Ryuichi S. Nagaosa

AIST, National Institute of Advanced Industrial Science and Technology, 1-1-1 Umezono, Tsukuba, Ibaraki 305-8560, Japan

ARTICLE INFO

Article history:
Received 27 December 2016
Received in revised form 3 March 2017
Accepted 21 March 2017

Keywords:
Heat exchanger
Computational fluid dynamics (CFD)
Pressure drop
Heat transfer coefficient
Flow transition
Open-source software

ABSTRACT

The purpose of this study is to quantify performances of a plain fin-and-tube heat exchanger based on a computational fluid dynamics (CFD) technique. This study employs a novel turbulence model-free approach to obtain exact numerical predictions and to avoid numerical uncertainties caused by the models. It applies three open-source software packages for conducting a series of CFD processes. The present CFD approach implements a total of 6.7×10^6 meshes to discretize the governing equations with the aid of viscous-layer meshes for fine-scale resolution, especially near the two flat-plate fins. The inlet air flow velocity is varied from 0.25 to 8.0 m s^{-1}, and the numerical results are compared with the laboratory measurements conducted under an equivalent experimental condition. The results of air flow visualizations show that the air flows can be categorized into three flow regimes, (1) a steady-state laminar flow, (2) an unsteady flow with periodic fluctuations, and (3) a turbulent flow with random fluctuations. This study identifies the critical Reynolds numbers for the transitions from (1) to (2) at $Re_D \approx 4000$, and from (2) to (3) at $Re_D \approx 6000$. The present numerical work also predicts the pressure drops and the heat transfer coefficients within an acceptable margin of errors. The fact demonstrates the potential usefulness and suitability of the present numerical approach for practical thermal engineering problems. It is concluded that the present simulation technique is beneficial to introduce for advanced design and optimization of heat transfer equipment with minimized numerical uncertainties.

1. Introduction

A heat exchanger is one of the important devices for transporting, converting, and utilizing thermal energy based on heat transfer between more than two fluids. The energy device has been applied to many industrial sectors, i.e., heating, ventilating, and air-conditioning (HVAC), chemical and mechanical engineering, automobiles, electronics, and smelting. Recently, energy conversion technologies have attracted considerable attention, as society is addressing climate change caused by combustion of fossil fuels, and the shift to net-zero energy. [1] Devices for energy conversion play an important role in the post fossil fuel era, [2] as the presence of renewable energies such as geothermal energy become larger in the global energy supply. [3] A study on design and optimization of the heat exchangers, therefore, has very long history in the field of environmental and energy engineering, and many reports on the issue have been published so far. Kays and London [4] have been focusing attention on the fundamental design of a compact heat exchanger, and numerous experimental data on the heat exchanger design are reported in their work. Comprehensive collections on heat exchanger design, optimization, and operation are summarized extensively by Shah and Sekulić [5].

There is growing interests in downsizing the devises, typically, from the order of 10 mm of tube diameter to smaller, such as several millimeters. [6] One of the motivations to downsize the devices is associated with very strong demands for miniaturization of products using heat exchangers, especially in the sector of HVAC. Downsizing the heat exchangers is advantageous to enhance flexibility for instrumentation and system optimization, as well as intensification for energy recovery processes. [7] Downsizing also contributes to increasing performance in energy-saving and environmentally-friendliness by reductions of electricity for running fans and blowers with intensifying density of thermal energy. Reduction of refrigerants such as hydrofluorocarbons (HFCs) to be charged in the heat exchangers, which is very urgent requirement for mitigation of the global warming, [8] can also be achieved effectively by downsizing the heat exchangers.

Designing and optimizing downsized heat exchangers require very sensitive validations of performances with difficult measurements of the pressure drops, temperature differences, and heat fluxes. Employing a numerical approach based on computational fluid dynamics (CFD) is assistive to optimize the design more

---

E-mail address: ryuichi.nagaosa@aist.go.jp

http://dx.doi.org/10.1016/j.enconman.2017.03.063

Nomenclature

Letters

AGF	area goodness factor, j/f (–)
A_C	frontal area (m^2)
A_W	surface area of test section (m^2)
C_P	specific heat ($\mathrm{J\,kg^{-1}\,K^{-1}}$)
D	tube diameter (m)
f	Fanning's friction factor (–)
g_i	vector of gravitational acceleration ($\mathrm{m\,s^{-2}}$)
H	heat flux ($\mathrm{W\,m^{-2}}$)
h_0	air-side heat transfer coefficient ($\mathrm{W\,m^{-2}\,K^{-1}}$)
j	Colburn j factor (–)
m	mass flow rate of the air, $\rho_0 U_{max} A_C$ ($\mathrm{kg\,s^{-1}}$)
M_0	molecular weight of the air ($\mathrm{kg\,mol^{-1}}$)
NTU	number of heat transfer unit (–)
P	pressure (Pa)
P_0	atmospheric pressure (Pa)
Pr	Prandtl number (–)
Ri	Richardson number, $\beta g D \Delta T / U_{in}^2$ (–)
Q	thermal energy exchanged across surface area of test section (W)
Re_D	Reynolds number, $U_{in} D/\nu$ (–)
$U,\ V,\ W$	air flow velocity ($\mathrm{m\,s^{-1}}$)
U_{in}	air inlet velocity ($\mathrm{m\,s^{-1}}$)
U_{max}	maximum velocity, U_{in}/s ($\mathrm{m\,s^{-1}}$)
St	Stanton number, $h_0/\rho_0 C_P U_{max}$ (–)
s	contraction ratio of cross sectional area (–)
T	temperature (K)
T_0	air inlet temperature (K)
T_{rms}	root-mean-square temperature fluctuation, $\left(\left\langle T^2\right\rangle - \langle T\rangle^2\right)^{1/2}$
T_W	temperature at fin and wall (K)
t	time (s)
$x,\ y,\ z$	coordinate (m)

Greek symbols

β	volume expansion coefficient (–)
ΔH	net heat flux brought into test section by convection ($\mathrm{W\,m^{-2}}$)
ΔP	pressure drop (Pa)
ΔT	temperature difference, $T_W - T_0$ (K)
ΔT_{LM}	logarithmic mean temperature difference (K)
Δt_0	targeted time step (s),
δ_{ij}	Kronecker's delta (–)
ν	viscosity (Pa s)
ρ	density of the air ($\mathrm{kg\,m^{-3}}$)
ρ_0	reference density of the air ($\mathrm{kg\,m^{-3}}$)
τ_W	viscous stress (Pa)

Superscripts

$*$	nondimensionalization by D/U_{in}

Subscripts

I	at inlet of test section
II	at exit of test section

efficiently. Numerous reports on experimental and numerical investigations on designing and optimizing the heat exchanger have been published for this reason. A very wide variety of fin geometries, for example, plain fin, [9–13] wavy fin, [14,15] corrugate fin, [16] louvered fin, [17–19] or a fin with winglets, [20,21] has been analyzed for advanced performance optimization. Many previous studies on designing and optimizing the heat exchangers based on numerical approaches have employed turbulence models. In particular, a two-equation model like a family of $k-\epsilon$ turbulence models together with the Reynolds-averaged Navier-Stokes (RANS) equations is often applied for investigating turbulent air flow and heat transfer characteristics. [22,23] The numerical strategies are thought to be practical for optimizing the heat exchanger configurations from a huge number of options of designing parameters, as saving CPU time and cost for computations are important for assisting quicker decision making. Indeed, introducing the turbulence models will be more important for practical designing and optimization of the devices with complex geometries. [24–26] The employment of turbulence models for heat transfer in the heat exchanger, on the other hand, has several issues to be addressed here, for example:

- A well-known commercial CFD software needs to switch calculation mode between "laminar" and "turbulent" for an adequate choice of viscosity model. Switching the calculation mode is generally difficult, since the critical Reynolds number of transition from laminar to turbulent is difficult to quantify.
- Applying the turbulence models to a spatially developed transient flow, or low-Reynolds number flow, is considered difficult, [27] and several improvements should be added to compensate the deficiency.
- No turbulence model is satisfactory to realize a fluid flow with flow separations and adverse pressure gradient, [28] which can be observed frequently in the air flow of a heat exchanger.
- The normal stress anisotropy close to a solid boundary may not be resolved properly in the two-equation turbulence models. Introducing the Reynolds-stress models (RSM), in which the transport equations of all the Reynolds-stress components are solved for a tensor expression of the eddy viscosity, is beneficial to overcome the deficiencies stated above. RSM, on the other hand, requires complex mathematical descriptions, and consequently, massive CPU resources for computations, and complexity of the numerical procedure

This study considers the application of a CFD technique without the turbulence models, which is often referred to as a direct numerical simulation (DNS) technique. The major motivation for employing the turbulence model-free approach is to demonstrate its applicability to engineering fluid flow and heat transfer problems. The CFD analyses with the turbulence models involve numerous uncertainties, for example, the analyses are often carried out based on low mesh resolution with lower-order discretization schemes. It should also be pointed out that effects of numerical viscosity caused by lower-order spatial and tempoeral discretization schemes are often involved. In addition, the turbulence-model parameters are not sufficiently optimized, and almost references use the standard parameters which are optimized in a turbulent flow in a simple fluid flow setup [29].

On the other hand, the turbulence model-free numerical approach provides very exact numerical solutions of the governing equations of fluid flow and heat transport in cases where a computational domain is resolved by sufficiently fine meshes. The turbulence model-free approach is beneficial to achieve very exact designing and optimization, since the approach does not involve any adjustable parameters, and numerical uncertainties caused by the turbulence models can be minimized. More specifically, the primary benefit of this study is that the numerical data reported in this study can be used as a reference for validating